ASIA TURBOMACHINERY & PUMP SYMPOSIUM
12-15 MARCH 2018
SUNTEC SINGAPORE

IMPELLER STALL INDUCED BY REVERSE PROPAGATION OF NON-UNIFORM FLOW

Shinji Iwamoto
Compressor Engineering Manager
Mitsubishi Heavy Industries Compressor Corporation
6-22, 4-Chome, Kan-On-Shin-Machi,
Nishi-Ku, Hiroshima, 733-8553, Japan

Shinji Iwamoto is presently Compressor Engineering Manager the within the Engineering and Design Division of Mitsubishi Heavy Industries Compressor Corporation (MCO) in Hiroshima, Japan. He is engaged in the design and development of centrifugal compressors in accordance with API Standard 617 for over 15 years. Mr. Iwamoto received B.S. and M.S. degrees in Mechanical Engineering from Kyushu University.

Yoshiaki Shoji
Compressor Engineer
Mitsubishi Heavy Industries Compressor Corporation
6-22, 4-Chome, Kan-On-Shin-Machi,
Nishi-Ku, Hiroshima, 733-8553, Japan

Yoshiaki Shoji is presently Compressor Engineer within the Engineering and Design Division of Mitsubishi Heavy Industries Compressor Corporation (MCO) in Hiroshima, Japan. He has been working on the design of compression systems and instrumentation, and in charge of the design of centrifugal compressors built to API standards for 6 years. Mr. Shoji received B.S degree in Information and Media Science from Tsukuba University.

Keizo Yoneda
Deputy General Manager
Global Marketing & Sales Division
Mitsubishi Heavy Industries Compressor Corporation
10F Shintamachi Bldg 5-34-6, Shiba,
Minato-ku, Tokyo 108-0014 Japan

Keizo Yoneda is representing as technical leader for specific projects and interfacing with customers during the requisition phase. His current responsibilities inclue providing technical solutions associate with turbo machinery and manage specific projects. He has been working on the design of compression systems built to API standards and applications engineering for over 15 years. Mr.Yoneda has B.S and M.S degrees in Mechanical Engineering from Fukui University.

Akihiro Nakaniwa
Research & Innovation Center Technology & Innovation Headquarter, Mitsubishi Heavy Industries, Ltd
6-22, 4-Chome, Kan-On-Shin-Machi,
Nishi-Ku, Hiroshima, 733-8553, Japan

Akihiro Nakaniwa is a project manager, Research & Innovation Center, Technology & Innovation Headquarters, Mitsubishi Heavy Industries, Ltd. He is engaged in aerodynamic designing and developing of centrifugal compressors for 15 years. Mr. Nakaniwa has M.S. degree from Kyushu University in Japan.

Bryan Barrington
Machinery Engineering
Lyondell Basell Global Engineering Services
Corpus Christi, TX 78410

Bryan Barrington is a 1994 graduate of Texas A&M University (BSME, Honors) and currently works as a Senior Advisor in LyondellBasell's corporate Machinery Engineering group. He provides support for turbomachinery at locations worldwide.

ABSTRACT

In the case of centrifugal compressors, minor non-uniform flow upstream of the impeller is induced by an asymmetrical configuration in the circumferential direction at the compressor suction casing. This non-uniform flow is transmitted to the impeller discharge, but this minor non-uniform flow does not usually cause an adverse effect on the impeller stage performance. However, we found this is amplified at the return channel due to flow separation at reduced flows (depending on return channel geometry), and the amplified non-uniform flow did induce impeller stall by reverse propagation from the return channel to the impeller.

These non-uniform flows caused a significant operating range reduction for a large flow coefficient impeller. The aerodynamics issues were mitigated using CFD analysis techniques, and eventually confirmed by the compressor performance during shop performance testing.

The OEM conducted the CFD analyses using two (2) return channel geometries with several CFD models to verify the effect of the return channel geometry on impeller stall and to

confirm the most suitable CFD modeling method for stall evaluation. Shop performance tests utilizing both return channel geometries were conducted and compared to the CFD analyses. These studies were conducted while collaborating with the end-user. The steady CFD calculation was conducted with frozen rotor interface between full annulus impeller and stator parts. The modeling of diffuser and return channel was varied as follows:

(1) 1-pitch model for the return channel with mixing plane at diffuser
(2) Full-annulus model for the return channel with a mixing plane at the diffuser
(3) Full-annulus model for the return channel without a mixing plane at the diffuser

From the above studies and the shop performance testing, it was confirmed that the proposed CFD modeling method could simulate the measurements taken during the shop performance tests and that the CFD modeling method utilized was key to properly evaluating stall phenomena.

INTRODUCTION

The impeller stage of an in-line centrifugal compressor consists of the suction casing, inlet guide vane (IGV), impeller, diffuser and return channel which are designed with the gas flow passage optimized for the rated point [Fig.1]. The role of the return channel located at the impeller discharge is to adjust the inlet angle of the gas flow for the next impeller, and to de-swirl the flow and convert the tangential momentum into static pressure. This adjustment is very important for impeller stage aerodynamic performance [Fig.2].

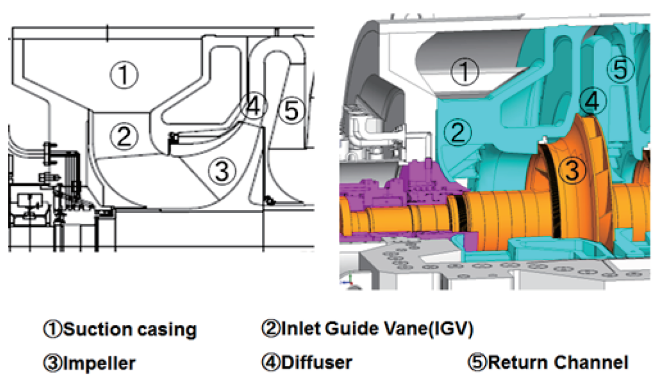


Fig.1 Compressor Stage Configuration

Shop performance testing of a propylene refrigeration compressor for an ethylene plant was conducted with a suction pressure of 5 psiA in accordance with ASME-PTC-10 Type-2 procedures. The design flow coefficient of 1st stage impeller was 0.15 and tip mach. number was 1.08 [Fig.3].

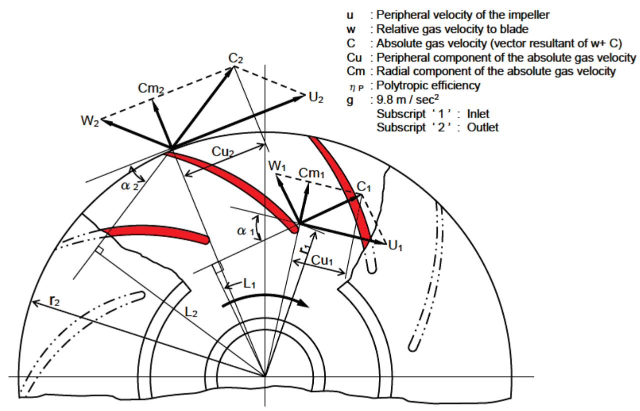
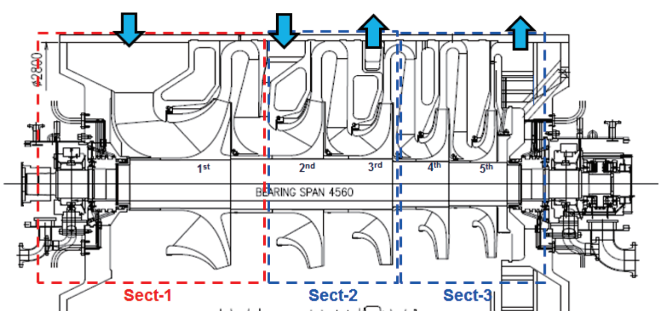


Fig.2 Aerodynamic Performance Principle



Compressor specification	
Propylene ref. compressor	11H-5S
Casing Inner Diameter	2800 mm
Bearing Span	4560 mm
MCR	3043 rpm
Compressor rated power	58582 HP

1st impeller specification at design point	
Design flow coefficient	0.15
Outer diameter	1640 mm
Vane height at exit	107 mm
Tip mach number	1.08
Design work coefficient	0.53

Fig.3 Propylene Ref. Compressor

During that test, compressor head of the first stage was decreased at approximately 13% higher flow than the predicted surge line, nevertheless compressor performance guarantee criteria was achieved at the guarantee operating point. Polytropic head dropped approximately 15% from the value at the stable operation point, but once head had dropped, stable operation in the lower flow range could be achieved without any surge phenomena [Fig.4]. As a cause investigation, CFD analyses with several models and tests were conducted to verify the cause of the stall phenomena and thereby identified the mechanism.

This paper introduces the mechanism of impeller stall induced by reverse propagation of non-uniform flow generated at the return channel which is observed when the design of the return channel is not suitable for reduced gas flow. In addition to that, a suitable modeling method for stall evaluation by CFD analysis will be suggested together with a key point to be carefully evaluated for return channel design.

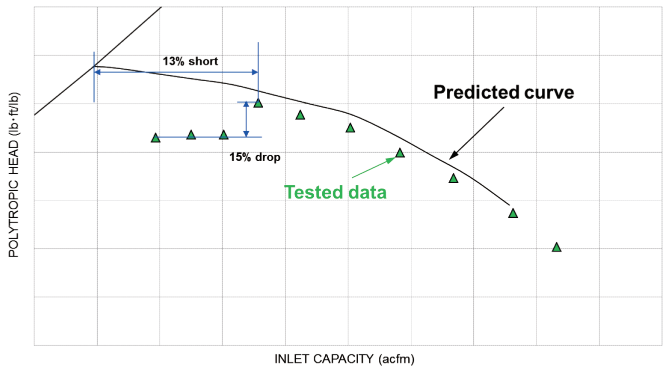


Fig.4 Shop Performance Test Result of Propylene Ref. Compressor

CFD ANALYSIS WITH SEVERAL MODELS

For the cause investigation, CFD analyses implementing several modeling methods were conducted as described in the following paragraphs. ANSYS-CFX code (0.1mm grid for near the wall surface) was used for the simulations.
The number of Node in each component is as follows.

- Suction Piping : 2.5 million
- Suction Casing : 14.4 million
- 1st stage (Impeller + Diffuser + Return channel) : 26.7 million
- Side Stream : 12.7 million
- 2nd stage (Impeller + Diffuser) : 12.1 million

ORIGINAL CFD MODEL

[Steady calculation with full-annulus model for impeller, 1-pitch model for return channel with Mixing Plane at diffuser]

[Fig.5] shows the original CFD model of the subject compressor. The CFD model consists of suction piping, suction casing, inlet guide vane, 1st stage impeller and the return channel. The Full-annulus model for the inlet guide vane & impeller and the 1-pitch model for the diffuser and return channel were applied. The frozen rotor interface between impeller and static parts such as "Inlet guide vane ⇔ Impeller", and "Impeller ⇔ Diffuser inlet" was used for simulation. At the diffuser outlet, a mixing plane technique was used. In case of frozen rotor interface, the relative orientation of the components across the interface is fixed. The two frames of reference connect in such a way that they each have a fixed relative position throughout the calculation. Effect of rotating (i.e. Centrifugal forces, etc.) on fluid is considered, but effect of the geometrical change (i.e. Pitch-change of impeller) is not considered. On the other hand, the mixing plane technique is an alternative to the frozen rotor model for modeling frame and/or pitch change. Instead of assuming a fixed relative position of the components, the mixing plane model performs a circumferential averaging of the fluxes through bands on the interface. Circumferential distortion is forced to average at the mixing plane. The mixing plane approach performs a mating between tangential averages on both sides of the interface between a rotor and stator which have a different number of blades. This approach is usually applied for a steady calculation with the pitch change interface.

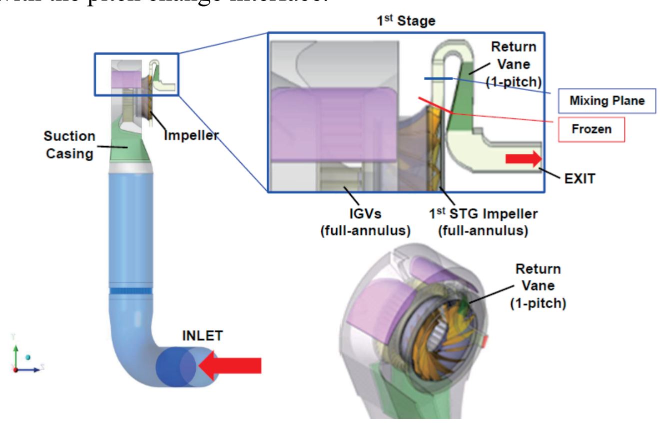


Fig.5 Original CFD Model

The CFD analysis result is shown in [Fig.6]. Due to an asymmetrical configuration in the circumferential direction at the compressor suction casing, non-uniform flow was generated upstream of the impeller, and this non-uniform flow transmitted to the impeller at low flow region. However, polytropic head continuously rose to the predicted surge line and the actual phenomena (such as a drop in polytropic head) could not be simulated by means of this analysis model.

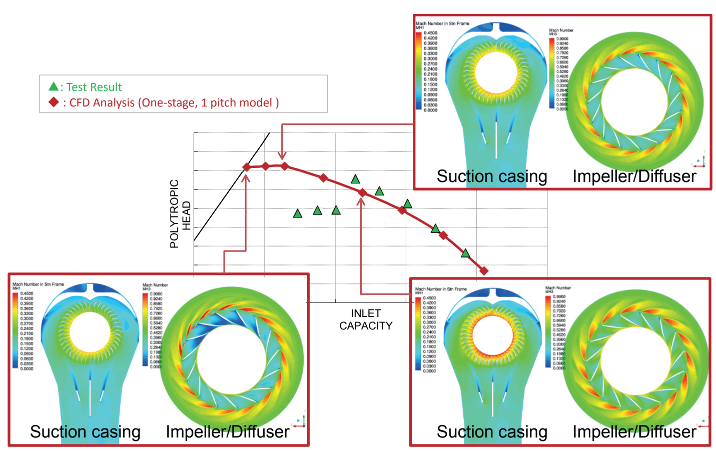


Fig.6 CFD Analysis Result (Original Model)

MODIFIED CFD MODEL-1
[Steady calculation with full-annulus model for impeller and return channel <u>with</u> Mixing Plane at diffuser]

[Fig.7] shows the CFD model (STEP-1) modified from the original one. Return channel and diffuser modeling was changed from 1-pitch to a full-annulus model, and the 2nd stage impeller was also modeled to confirm the influence of the 2nd stage condition to the upstream stage. However, there was no major difference from the calculation results of the original model [Fig.8].

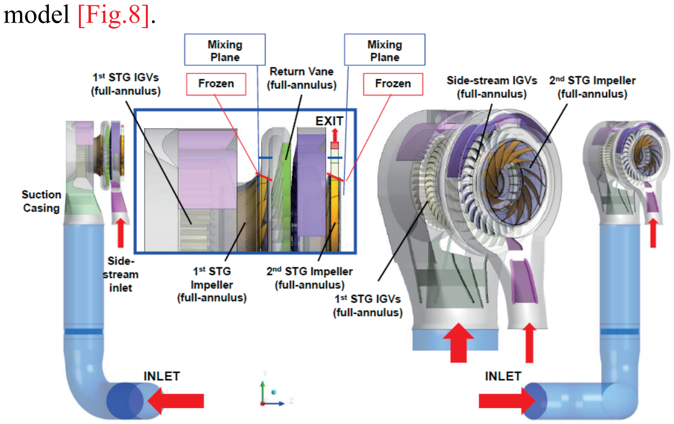


Fig.7 Modified CFD Model-1 (STEP-1)

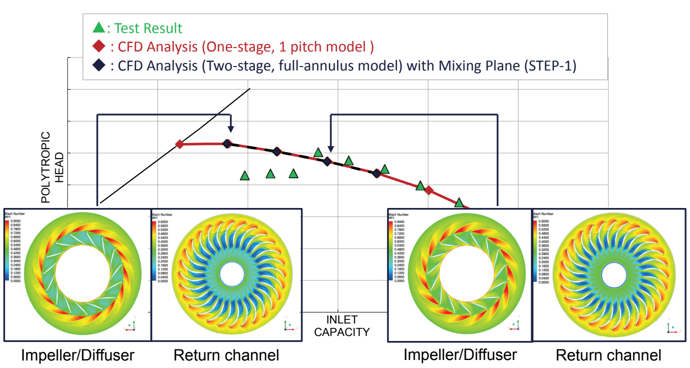


Fig.8 CFD Analysis Result (Modified Model-1 STEP-1)

Therefore, as an alternative approach, the following STEP-2 was considered in addition to STEP-1 [Fig.9].

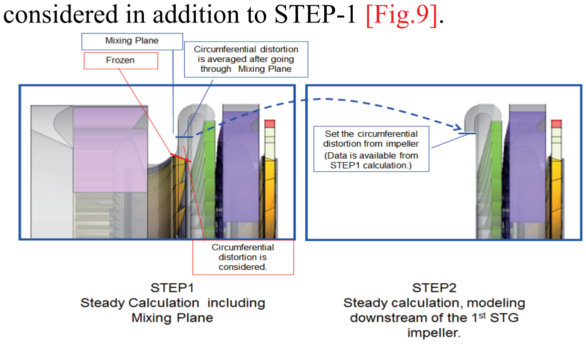


Fig.9 Modified CFD Model-1 (STEP-2)

STEP-1) Steady calculation from compressor suction casing to 2nd stage diffuser outlet with Mixing Plane at 1st stage diffuser inlet.

STEP-2) Steady calculation from 1st stage diffuser inlet and 2nd stage diffuser outlet. The circumferential distortion at 1st stage diffuser inlet which calculated by STEP-1 was set as inlet boundary condition.

As a first step, steady calculation of the 1st stage (from suction casing to impeller outlet) was conducted to confirm circumferential distortion at the diffuser inlet. Then, as the second step, calculated circumferential distortion was set as boundary condition at the diffuser to confirm the effect of circumferential distortion for the return channel. Basically, when the mixing plane is applied, circumferential distortion is averaged after going through the mixing plane, and circumferential distortion at the impeller outlet could not be transmitted to the return channel. However, this technique can allow transmittal of the circumferential distortion to the return channel. [Fig.10], [Fig.11] and [Fig.12] illustrate the CFD analysis results.

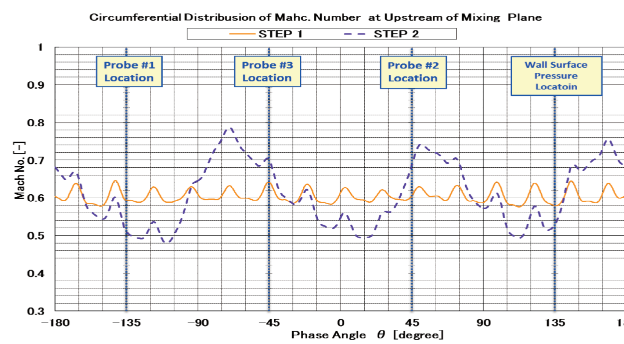


Fig.10 Circumferential Distortion at Diffuser Inlet (upstream of Mixing Plane)

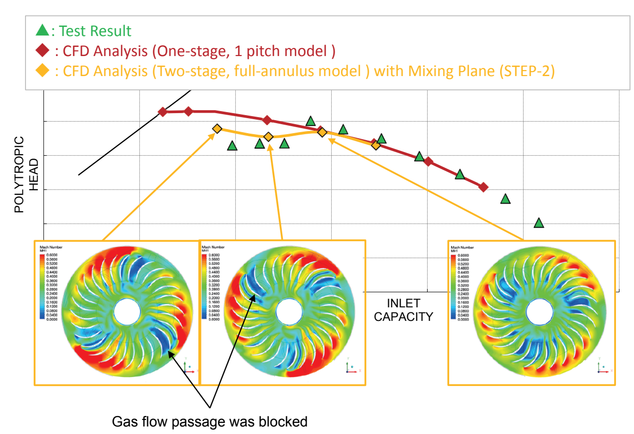


Fig.11 CFD Analysis Result (Modified Model-1 STEP-2)

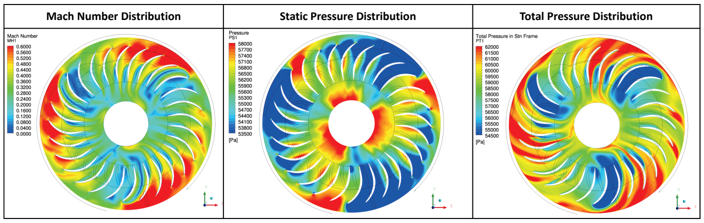


Fig.12 CFD Analysis Result (Modified Model-1 STEP-2)

Circumferential distortion at the Mixing Plane was minor for STEP-1, but it was amplified by the calculation of STEP-2. Color contour shows the Mach Number at the return channel. As shown on this figure, blockage of gas flow passage was confirmed at the return channel at the reduced flow region and incidence of polytropic head reduction could be simulated at approximately the same flow as the shop test although there is a slight difference in the absolute value of the polytropic head. As shown by the meridional vector with radial velocity color at the reduced flow region in [Fig.13], reverse flow was confirmed at the return bend but not confirmed at impeller since a Mixing Plane had been applied for the diffuser.

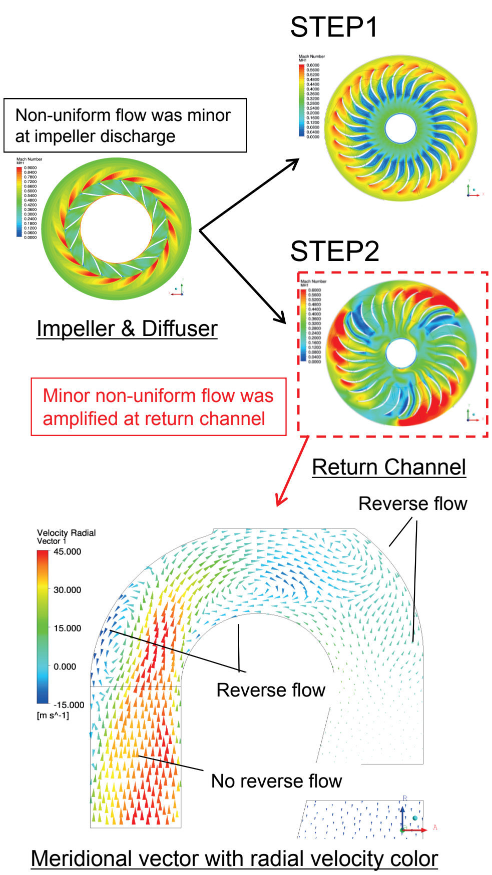


Fig.13 Analysis Result (CFD Model-1)

MODIFIED CFD MODEL-2

[Steady calculation with full-annulus model for impeller and return channel without Mixing Plane at diffuser]

[Fig.14] shows the modified CFD model-2. Mixing plane at diffuser inlet was removed from CFD Model-1. In the case of the modified model-1, reverse non-uniform flow developed at the return channel was not propagated to upstream of the diffuser and impeller since non-uniform flow is averaged by Mixing Plane. In contrast, in the case of the CFD model-2 (since Mixing Plane at diffuser is removed), non-uniform flow can be transmitted to upstream of the diffuser and impeller and influence on compressor performance due to reverse propagation of non-uniform flow generated at return channel can be simulated [Fig.15].

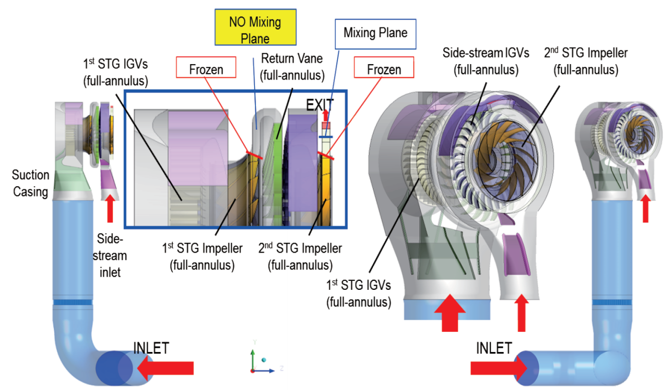


Fig.14 Modified CFD Model-2

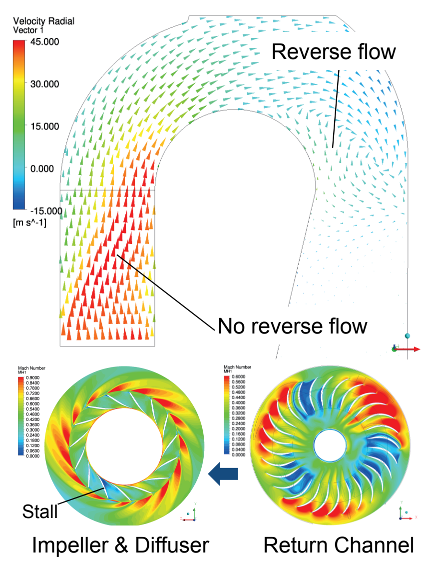


Fig.15 Analysis Result (CFD Model-2)

[Fig.16] shows the results of the CFD analysis. The polytropic head decreased more than the modified model-1 and it was more conservative than the test results. [Fig.17] shows the pressure loss coefficient and pressure recovery coefficient at return channel. The pressure loss coefficient at return channel was not so different among the analysis results of three models. However, the large difference for the pressure recovery coefficient at return channel was confirmed in the CFD analysis due to the difference of flow pattern at return channel.

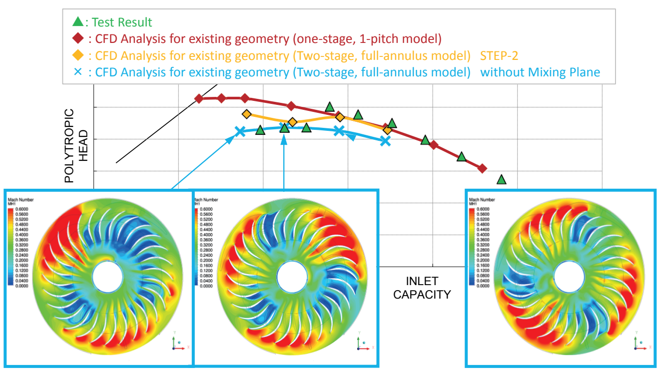


Fig.16 CFD Analysis Result (Modified Model-2)

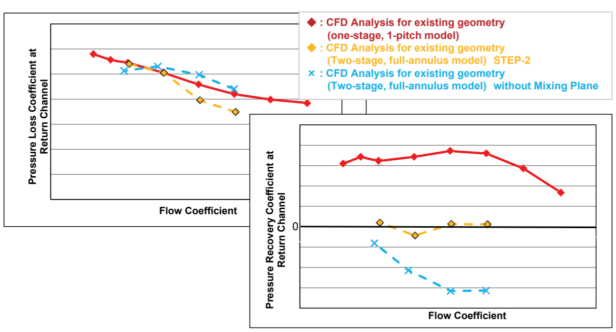


Fig.17 Pressure Loss Coefficient and Pressure Recovery Coefficient at Return Channel

In addition to that, as shown on [Fig.18], the degree of non-uniform flow was increased by the effect of removing the mixing plane, and consequently non-uniform flow developed at the return channel traveled to upstream of the impeller and deteriorated the performance of the impeller. Considering the above, the actual trend is better simulated by the CFD analysis without mixing plane (Model-2) but polytropic head calculated by Model-1 better matched the test results.

STALL MECHANISM INDUCED BY NON-UNIFORM FLOW GENERATED AT RETURN CHANNEL

In order to clarify the mechanism of this phenomenon, OEM conducted the special measurements such as total pressure, static pressure and pressure fluctuation measurement at return bend during shop performance test [Fig.19].

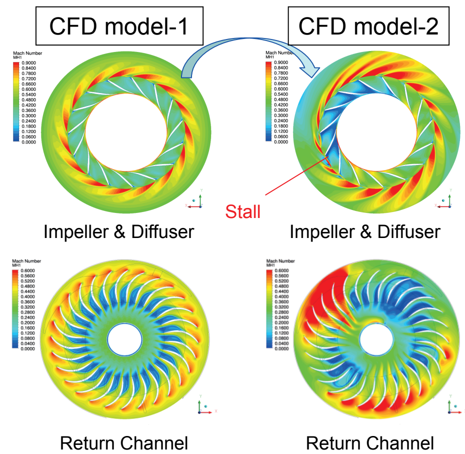


Fig.18 Mach Number Color Contour at Impeller & Return Channel

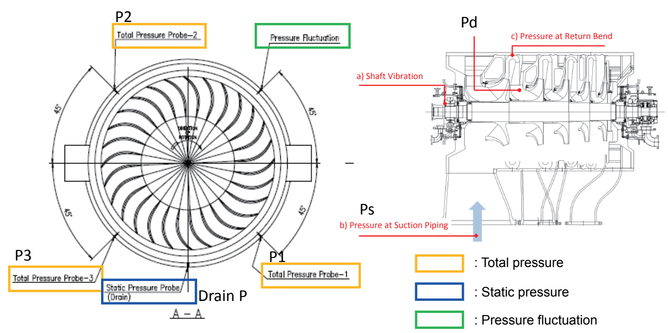


Fig.19 Special Measurement during Shop Performance Test

As shown on [Fig.20], pressure fluctuation was observed at return bend after head was decreased, but the pressure fluctuation was not observed at suction line. At the more reduced flow range, pressure fluctuation was observed at suction line as same as return bend.

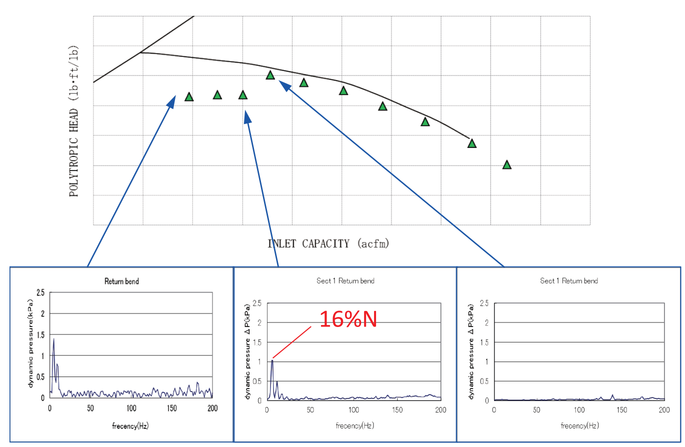


Fig.20 Pressure Fluctuation at Return Bend

[Fig.21] shows pressure ratio between compressor suction piping and return bend, or return channel outlet. As shown on this figure, total pressure for both return bend and return channel outlet was reduced, but reduction rate of return channel outlet was larger than that of return bend. That means large pressure loss would be generated through return channel.

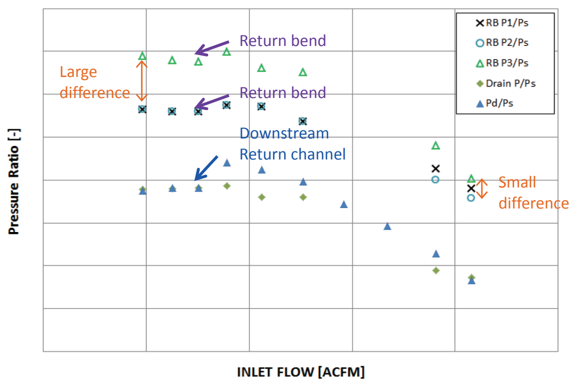


Fig.21 Pressure Ratio between Suction Piping and Return bend or Return Channel Outlet

From the above special measurement results and CFD analysis results, the OEM hypothesized that the scenario for the mechanism of the observed phenomena during the shop test progressed as follows: [Fig.22].

(1) When gas flow was reduced, flow separation was generated at the gas flow passage in the outlet of the return channel due to the large velocity reduction.
(2) The gas flow passage was blocked by the generated separation at the return channel and the gas flow was suddenly reduced. And pressure at outlet of return channel was decreased by the large pressure loss through the return channel.
(3) The local backpressure was increased around the return bend due to the blockage at the return channel.
(4) The local backpressure propagates to the impeller trailing edge, and eventually causes an impeller local stall.
(5) This mechanism caused the polytropic head reduction.

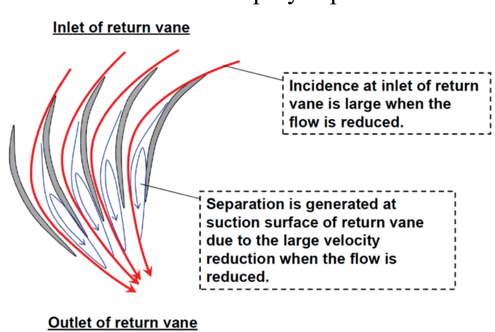


Fig.22 Scenario of Observed Phenomenon

Considering the above, since the primary cause of the polytropic head reduction was flow separation at the outlet of the return channel due to the large velocity reduction, the flow area of the outlet of the return channel was reduced to eliminate the large velocity reduction [Fig.23].

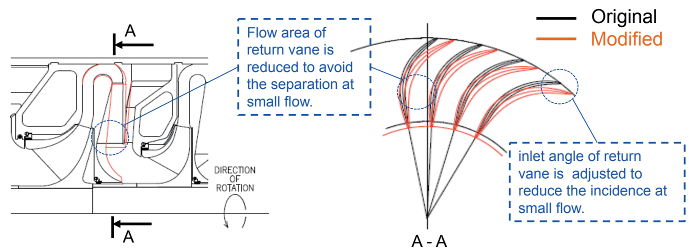


Fig.23 New Return Channel Geometry

[Fig.24] shows the test result and the CFD analysis result with the new geometry of the return channel. As shown in this figure, after the new geometry was applied at the return channel, generation of non-uniform flow and separation was significantly reduced and the originating point of polytropic head reduction was shifted to the reduced flow region. CFD analysis by Model-2 agreed well with the test result.

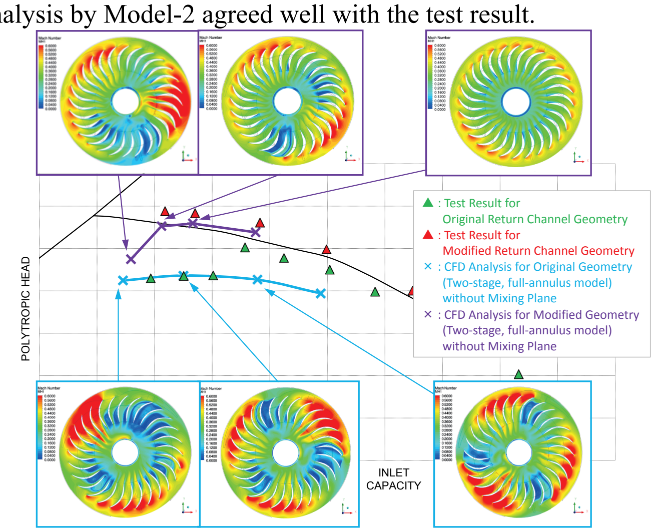


Fig.24 Comparison between Test Result & CFD Result with New Return Channel Geometry

ROOT CAUSE VERIFICATION BY COMPONENT TEST MACHINE

As an additional cause investigation, several verification tests were conducted by means of component test facility (single stage performance test facility) in our laboratory which consists of movable inlet guide vane, impeller, diffuser, return bend and return channel [Fig.25]. In the verification tests, as a first step, impeller stage performance with optimum configuration of static parts such as inlet guide vane angle, diffuser ratio and return channel geometry was confirmed (hereinafter called

"Original Case"). After that, many different test cases were carried out by changing the static parts (such as the suction casing, the angle of the IGV, the resistance at the discharge line, and the return channel geometry) in order to evaluate the influence on the compressor performance by following items:

Case-1) Non-uniform flow upstream of impeller
Case-2) Non-uniform flow downstream of return channel
Case-3) Return channel geometry

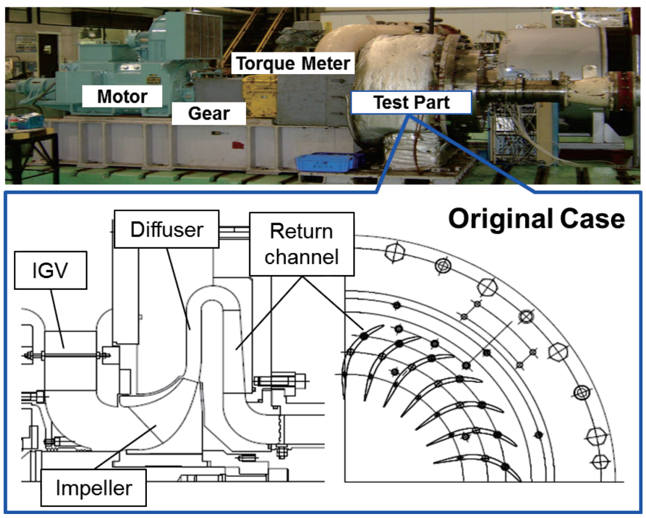


Fig.25 Single Stage Performance Test Facility

[Fig.26] shows the arrangement of test case-1 and case-2. In order to generate non-uniform flow upstream of the impeller, a perforated plate on the top half of the IGV was installed (Case-1a). In addition to that, the angle of the inlet guide vane was set to simulate the deviated flow toward to the impeller center (Case-1b). Furthermore, in order to simulate the non-uniform downstream of the return channel due to reduced side stream flow etc, exit of the return channel was half covered by a perforated plate (Case-2).

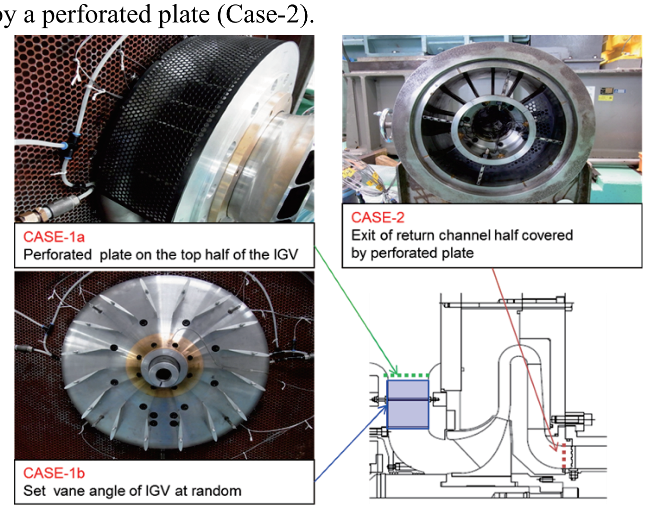


Fig.26 Non-uniform Flow Test on Upstream of Impeller

[Fig.27] shows the test arrangement of Case-3. When designing a return channel, the inlet area of the return channel is designed as per the upstream impeller and the outlet of the return channel is designed as per the downstream impeller. Consequently, if a large flow type impeller is installed downstream, the ratio of the inlet and outlet area of the return channels is increased. At reduced flows, this increase may cause a large gas velocity reduction and result in flow separation and pressure loss. Therefore, in order to evaluate the influence on compressor performance of a wide area at the outlet of the return channel, CFD analysis and verification test Case-3 was conducted.

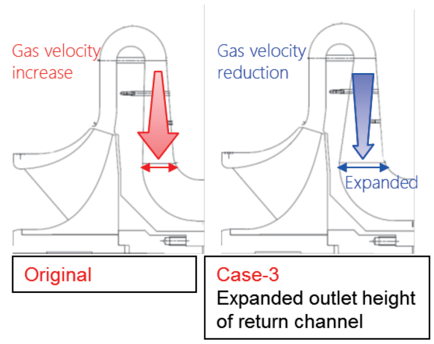


Fig.27 Wider Outlet Area of Return Channel

[Fig.28] shows the CFD analysis result of the Case-3. When gas flow was reduced, flow separation was generated at the gas flow passage, and this phenomena is well matched with the mechanism which shown on Fig.22.

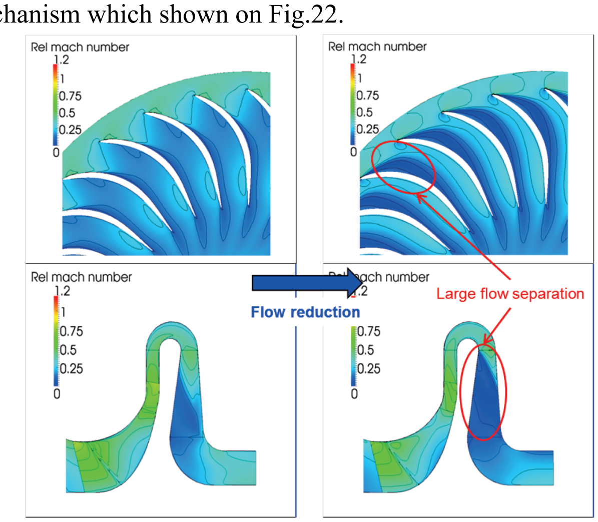


Fig.28 CFD Analysis Result of the Case-3

[Fig.29] shows the performance test results of the original case as well as Cases-1~3. The horizontal axis shows the normalized flow coefficient, and the vertical axis shows the normalized pressure coefficient of the impeller stage (from suction piping to return channel outlet). Comparing the original condition and Case-1a & 2, the compressor performance was almost the same with no incidence of pressure decrease being observed at the reduced flow region. In case of Case-1b, the pressure coefficient was decreased slightly compared to the original case, but the trend was the same as the original case. From these results, it was confirmed that the actual phenomenon of pressure drop experienced during the shop test of the propylene refrigeration compressor would not be induced by the non-uniform flow created intentionally at upstream of impeller as well as downstream of return channel. In contrast, in case of Case-3, the pressure coefficient dramatically decreased at the reduced flow region, and the same phenomenon as the shop performance test of the propylene refrigeration compressor was simulated. [Fig.30] shows the pressure loss coefficient and the pressure recovery coefficient at return channel. The pressure loss coefficient of Case-3 increased from the original, and pressure recovery coefficient of Case-3 dramatically decreased at the reduced flow region.

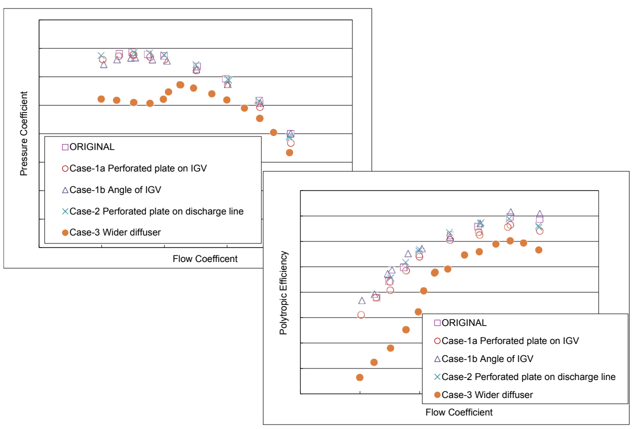


Fig.29 Single Stage Performance Test Result

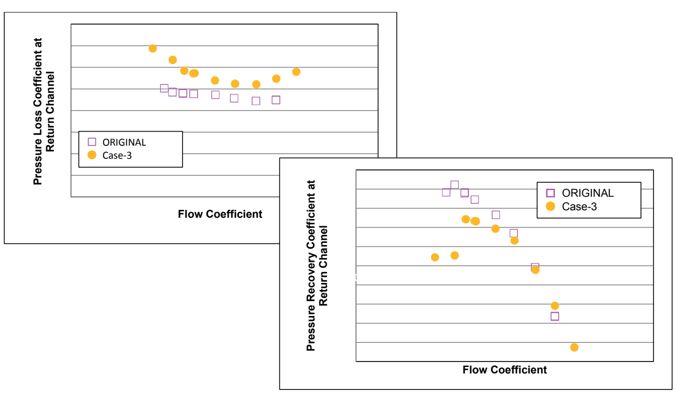


Fig.30 Pressure Loss Coefficient and Pressure Recovery Coefficient at Return Channel

[Fig.31] shows the pressure coefficient at diffuser outlet. As shown on this figure, pressure coefficient of diffuser outlet was also dropped at reduced flow range as same as pressure coefficient of impeller stage. [Fig.32] shows the FFT analysis results of the pressure fluctuation measured at the diffuser outlet. In the case of the original configuration, the non-synchronous component at the low frequency range was not observed at the reduced flow region. However, in the case-3, the non-synchronous component at the low frequency range due to stall appeared.

From above test results and the CFD results, we concluded that the root cause of the actual phenomenon of the pressure drop was "impeller stall" induced by reverse propagation of non-uniform flow generated at the return channel due to application of a return channel dimension that did not match the upstream impeller.

The OEM is now conducting additional component tests with alternative return channel geometry in order to confirm whether the stall phenomenon can be mitigated by minimizing the generation of flow separation at the return channel outlet due to large velocity reduction. We expect this test result will generate useful design criteria for return channel geometry.

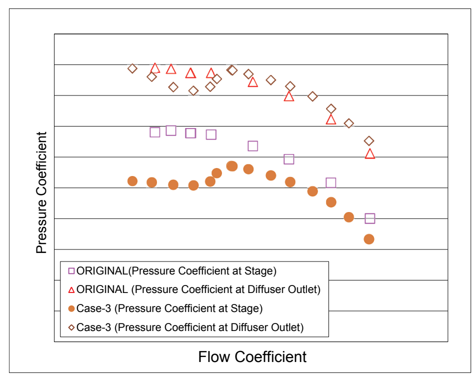


Fig.31 Pressure Coefficient at Diffuser

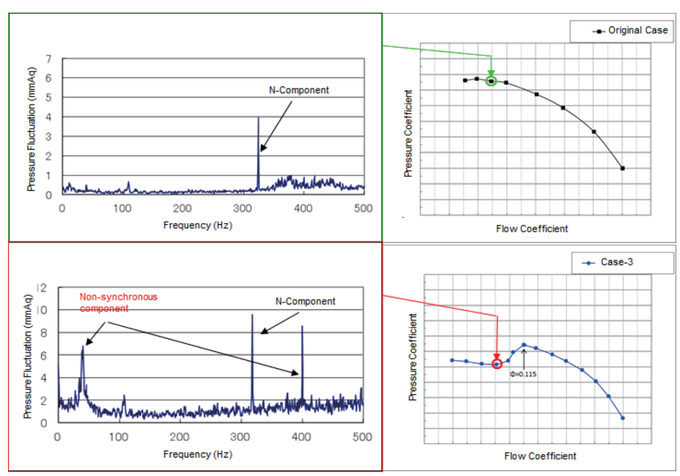


Fig.32 FFT Analysis Result of Pressure Fluctuation

CONCLUSIONS

The OEM conducted several CFD analyses as well as verification tests in order to evaluate the root cause of the stall phenomenon while collaborating with the end user. From the above studies and verification tests, it was confirmed that the root cause of this phenomenon was "impeller stall" induced by reverse propagation of non-uniform flow generated at the return channel due to application of a return channel dimension that did not match with the upstream impeller. It was also confirmed that the combination of following two (2) CFD modeling methods could best simulate the actual stall phenomena originating point and absolute value of polytropic head:

Full-annulus model for the impeller/ return channel with a mixing plane at the diffuser with following two (2) steps;

(1) Steady calculation from compressor suction casing to 2nd stage diffuser outlet with Mixing Plane at 1st stage diffuser inlet.

(2) Steady calculation from 1st stage diffuser inlet and 2nd stage diffuser outlet. The circumferential distortion at 1st stage diffuser inlet which calculated by STEP-1 was set as inlet boundary condition.

Generally, impeller stall is induced by non-uniform flow generated upstream of the impeller. However, as introduced in this paper, the geometry downstream of the impeller also affects impeller stall. Therefore, the design procedure for components downstream of the impeller especially velocity reduction through return channel is important, and non-uniform flow should be carefully evaluated when completing compressor aerodynamic performance design.

REFERENCES

1. API Standard 617, Seventh Edition, July 2002 "Axial and Centrifugal Compressors and Expander-compressors for Petroleum, Chemical and gas Industry Service".
2. ASME PTC10-1997 "Performance Test Conde on Compressors and Exhausters"
3. S. Iwamoto, S. Tokuyama, A. Nakaniwa, K. Yoneda, H. Higuchi, B. Barrington, September 2016 "Impeller Stall Induced by Reverse Propagation of Non-Uniform Flow Generated at Return Channel" of 45th Turbomachinery & 32nd Pump Symposia.

ACKNOWLEDGEMENTS

The authors would like to acknowledge Eric Krivitzky, Colin Osborne and George Zitka of ConceptsNREC for their participation and support in developing and interpreting the CFD models.